Purdue University
Purdue e-Pubs

International Compressor Engineering Conference School of Mechanical Engineering

2018

# Three-dimensional CFD Simulation of a Roots Blower for the Hydrogen Circulating Pump

Yian Zhang
*Xi'an Jiaotong University, China, People's Republic of*, iamspecial1994@stu.xjtu.edu.cn

Ying Zhao
*Xi'an Jiaotong University, China, People's Republic of*, zy18706704461@stu.xjtu.edu.cn

Xueyuan Peng
*Xi'an JiaoTong University, China, People's Republic of*, xypeng@mail.xjtu.edu.cn

Follow this and additional works at: https://docs.lib.purdue.edu/icec

Zhang, Yian; Zhao, Ying; and Peng, Xueyuan, "Three-dimensional CFD Simulation of a Roots Blower for the Hydrogen Circulating Pump" (2018). *International Compressor Engineering Conference.* Paper 2625.
https://docs.lib.purdue.edu/icec/2625

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.
Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at https://engineering.purdue.edu/Herrick/Events/orderlit.html

# Three-dimensional CFD Simulation of a Roots Blower for the Hydrogen Circulating Pump

Yian ZHANG[1], Ying ZHAO[1], Xueyuan PENG[1,2]*

[1]Xi'an Jiaotong University,
Xi'an, Shaanxi, China

[2]National Engineering Research Center of Fluid Machinery and Compressors,
Xi'an, Shaanxi, China
E-mail: xypeng@mail.xjtu.edu.cn

* Corresponding Author

## ABSTRACT

Fuel cell vehicle (FCV) is more advantageous compared to a gasoline-powered internal combustion engine based vehicle and is a new and popular way to solve the environmental problem. Hydrogen circulating pump is an essential assembly unit for the fuel cell, which is used in FCV. Roots blower is employed in this field. Hydrogen has a low molecular weight, which means it is easy to leak through the clearances and hard to preserve. It increases the difficulty of design and operation.

In this paper, the performance parameters of a three-lobe roots blower were studied. A multi-scale model was used since the clearance, compared with the rotor diameter, is quite small and easy to induce gas leakage. The long pipes and the throttle values were included in this model. To improve the accuracy and the mesh quality, the working chambers and clearances were represented by highly deforming dynamic structured grids and the grids of the clearances had a high density. The suction and discharge ports were meshed by static unstructured grids. The interfaces were used between the two rotors and between the rotors and the suction and discharge ports.

The performances were compared at three different rotational speeds of 4500 rev/min, 6000 rev/min and 8000 rev/min using hydrogen as the working fluid in this numerical model with almost the same inlet and outlet pressures. Meanwhile, the effect of the leakage was also analyzed using two different clearances, which had an important influence for the designing and using of the roots blower.

## 1. INTRODUCTION

At present, global energy and environmental issues are increasingly serious and some countries are seeking solutions desperately. The automotive industry also responds to this. In recent years, new energy vehicles such as electric vehicle (EV), hybrid electric vehicle (HEV) and fuel cell vehicle (FCV) are developed increasingly and put in vast efforts and stepping into the market, which have become an upwind trend. Among them, FCV not only can achieve complete replacement of fuel, but also has the advantages of "zero emissions", high energy conversion efficiency, and diverse fuel sources. Thus it is considered to be one important direction of sustainable automobiles development of dealing with global energy and environmental issues. Nowadays, many countries are making greater efforts to develop FCV. Companies such as Honda, Toyota, General Motors, Mercedes-Benz, Ford and Hyundai have all developed FCV, conducted demonstration operations, and entered preliminary application stages. However, the research and development of key components, vehicle integration and durability are not developed in the same way. For example, in the hydrogen circulation device of the accessory system, Park Company in the United States developed a hydrogen circulation pump that can be used in different FCV. The major automobile companies have also developed corresponding hydrogen circulation devices which are used in fuel cell engines, like screw and centrifugal devices. The development of fuel cells is imminent.

The roots blower is a positive-displacement double-rotor gas delivery device that can be used as a blower and vacuum pump. It is simple in structure, easy to maintain, does not require internal lubrication, and the displacement is almost constant in the pressure range of use. The leakage problem of the rotating machinery is also a concern that researchers have focused on. Panpan Song et al. (2017) mainly focus on the radial leakage flow pattern of the axial clearance on a scroll-type expander for a small scale ORC system with R245fa. Rainer Andres et al. (2016) show workflow and results of the Screw Expander SE-51.2. Most of these simulations use air as the working medium. Hydrogen, due to its low relatively molecular weight, easily leaks during operation. The CFD method has developed rapidly in these years, with low cost, capable of simulating more complicated or ideal processes. It has also been widely used in rotary machines, such as small-scale ORC expanders (Bianchi et al., 2017), sliding vane rotary machines (Bianchi et al., 2017), screw compressors (Rane et al., 2013), and oil-free scroll expander (Suman et al., 2017), rolling piston compressor (Hu et al., 2017) and scroll compressor (Chen et al., 2016). Nevertheless, the simulation in this method of roots blower is not as many as others, and the model applied in those simulations just had some simplified inlet and outlet ports and short pipes. This situation was improved in this paper.

In this paper, the internal flow field and performance of a roots blower applied in a hydrogen circulation system were numerically investigated using the CFD method. To make the simulation more accurate and closer to reality, a model with lengthened pipes and throttle valves installed on the inlet and outlet pipes was employed. The working chambers were meshed by dynamic structured grids. The pipelines and throttle values were meshed into static unstructured grids. The hydrogen was used as the working medium, and the performances at three speeds and under different gaps were compared. The numerical simulations discussed in this paper can provide guidance for future experimental studies.

## 2. GEOMETRY

The flow domain of a roots blower is decomposed into three sub-domains, namely inlet port, rotor flow domain (namely working chambers) and outlet port. The roots blower modeled in this paper was a three-lobe straight blade type. The profile line used was an asymptote type, with a higher area utilization factor. The rotor profile is shown in Figure 1, and the dimension settings are shown in Table 1. Traditional roots blower studies have neglected the influence of the pipeline on the internal flow field, generally given constant pressures at short inlet and outlet ports; but in the actual process, the roots blower suction and discharge are conducted through the pipeline, and there are inevitable throttling devices outside the inlet and outlet ports. For this reason, the pipelines were lengthened and the throttle values were added in the pipes on the basis of the previous simple model. The inlet and outlet pipeline model is shown in Figure 2. The projected inlet pressure was 0.23 MPa and the outlet one was 0.3 MPa. Through trial and error, the diameters of the inlet and outlet throttle valves were determined. But the pressures were changed slightly, which will be seen in the fifth section. Detailed dimensions are shown in Table 2.

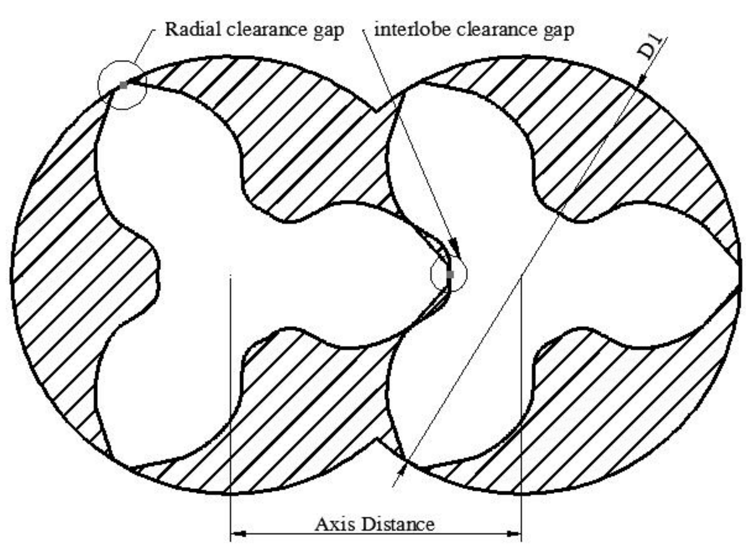


**Figure 1:** Profile

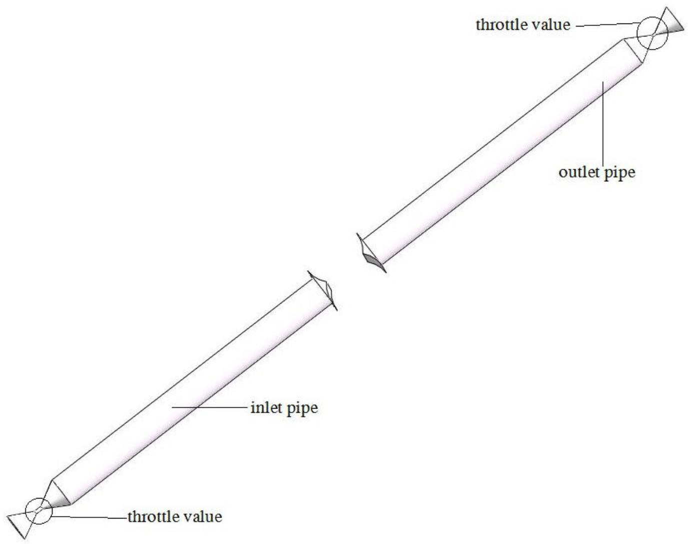


**Figure 2:** The inlet and outlet pipes and throttle values

**Table 1:** The dimension settings

| Outer Diameter D1 (mm) | Axial Distance (mm) | Rotor Length L (mm) | Interlobe Clearance (mm) | Radial Clearance (mm) |
|---|---|---|---|---|
| 75 | 50 | 50 | 0.08 | 0.04 |

**Table 2:** The diameters of the throttle values

| Rotational Speed (rev/min) | Inlet Throttle Diameter (mm) | Outlet Throttle Diameter (mm) |
|---|---|---|
| 4500 | 2.8 | 1.2 |
| 6000 | 4 | 2.6 |
| 8000 | 6 | 4 |

## 3. MESHING

The working chambers of the roots blower are constantly merged and separated, and the gap area is also constantly moving, deforming, disappearing and appearing, and since the size of the roots blower gap and the size of the working chamber are different by three or four orders of magnitude, high demands are required on the quality of the grids. Therefore, using high-quality discrete methods to make CFD simulations with more accurate results is crucial.

### 3.1 Principle of Grid Generation

There are many complex areas whose boundaries are not exactly in line with the existing various coordinate systems. Therefore, a method of calculation can be used to create a coordinate system whose axes are adapted to the boundary of the object being calculated. The coordinate system is called the body-fitted coordinate system. Body-fitted coordinate system schematic is shown in Figure 3.

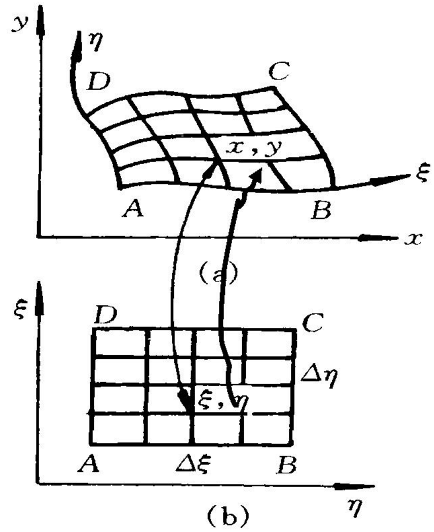


**Figure 3:** Body-fitted coordinate system schematic

The grid generation is to select the solution area corresponding to the complex area on the physical plane on the calculation plane, and find out the corresponding relationship of the internal nodes of the two areas, like equation (1) or its inverse function.

$$x = x(\xi,\eta), y = y(\xi,\eta) \tag{1}$$

### 3.2 Meshing

Since the shape of the chambers change with rotation, the dynamic mesh was selected. The rotary process was divided into several parts according to the time step, which means that every time step corresponded to a position of the rotors. Then the fluid domain of every position was meshed into the static structured grids through the body-fitted coordinate. When the time step was smaller, the number of the positions was larger. When CFX was calculating, with time step going by, the grids of every position was called and calculated, forming dynamic grids.
The fluid shape of the inlet and outlet pipelines and the throttle valves did not change with the rotation of the rotor, so they were meshed into static unstructured grids by Workbench directly.

## 4. SETUP

Both the rotor fluid grids and the pipeline grids were imported into CFX. Numerical grids for rotor domains and roots blower ports domains were connected through non-conformal conservative interfaces in the solver selected for CFD calculation.

### 4.1 Boundary Conditions

The inlet and outlet boundaries are shown in Figure 4. Both the inlet and outlet boundary types were opening, which prevented the backflows. The inlet and outlet temperatures were 293.15 K and the pressures were 0.24 MPa; the other boundary conditions were wall boundary conditions. Three monitor points were set and shown in Figure 4. The first one was on the rotor fluid domain, the second one was on the outlet pipe and close to the throttle value, and the third one was on the inlet pipe and also close to the throttle value. The absolute pressures of these three points were obtained. The location of the monitor points are shown in Figure 4.

### 4.2 Initial Conditions

Reasonable initial conditions can effectively speed up the convergence of calculation results. The initial temperatures and initial pressures of the rotor fluid, inlet pipeline, and outlet pipeline were set as 293.15 K and 0.24 MPa, respectively.

### 4.3 Turbulence Model

SST model was employed in this simulation. The k-ω equation based on the SST model takes into account the transmission of turbulent shear stress and can accurately predict the flow separation and the amount of fluid under negative pressure gradient conditions. It does not over predict the eddy viscosity. This makes the SST k-ω model more accurate and reliable than the standard k-ω model in a wide range of flow areas.

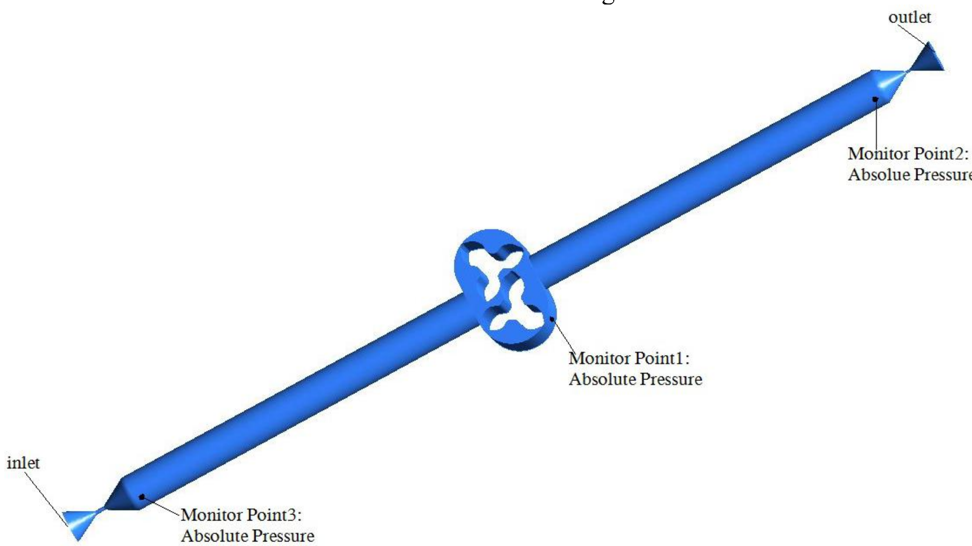


**Figure 4:** Inlet and outlet boundaries and the monitor points

## 5. RESULT AND DISCUSSION

The numerical simulation of the internal flow field of the roots blower was a transient simulation. When convergence, the calculation result would not approach a certain value but would change with time. For periodical numerical simulation, the basis for the transient calculation convergence judgment can be started from the following points: the residual values of convergence in each time step needs to be lower than RMS residual, each parameter changes cyclically, and inlet and outlet average flow relative errors are less than 2%.

### 5.1 Results and Discussion for Various Speeds

The results of the simulation include the pressures results, mass flow results and torque results. These results are analyzed and discussed.

5.1.1 Pressure: As mentioned above, the prospective pressures were that the inlet pressure was 0.23 MPa and the outlet pressure was 0.3 MPa. But the real results had slightly differences from the ideal ones by using throttle values rather steady pressures. The actual monitor point pressures are shown in Table 3. It can be seen that the pressures ratio of the three conditions were almost the same, which indicated that although there were some errors between the ideal pressures and the real pressure, the results and comparison below were still valid. The trend of the monitor point 1 is shown in Figure 5. It shows the change of pressure in the 6000 rev/min.

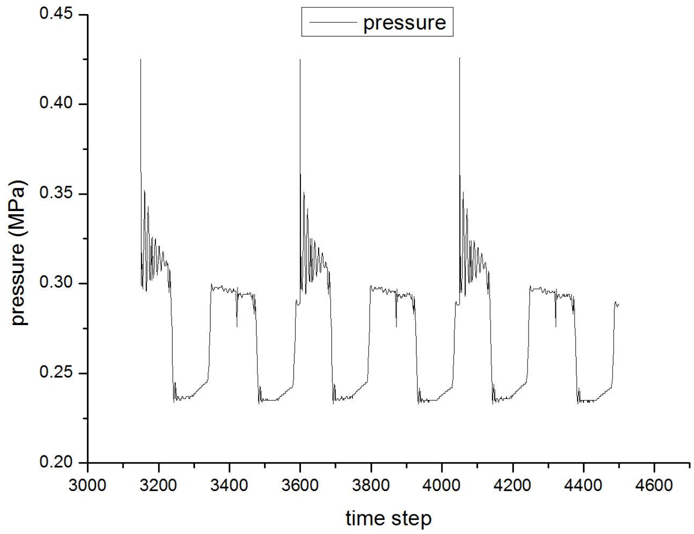


**Figure 5:** Pressure of point 1

**Table 3:** Monitor point pressures

| Rotational Speed (rev/min) | Point 3 (Absolute Pressure of the Inlet Pipe, MPa) | Point 2 (Absolute Pressure of the Outlet Pipe, MPa) | Pressure Ratio |
|---|---|---|---|
| 4500 | 0.2328 | 0.2953 | 1.268 |
| 6000 | 0.2336 | 0.2965 | 1.269 |
| 8000 | 0.2333 | 0.2957 | 1.267 |

5.1.2 Mass flow: Periodic change of the mass flow in 6000 rev/min is shown in Figure 6. Other conditions had the same trends. The last complete period was selected and its average flow was calculated. The results are shown in Table 4. When the relative error of the mass flow was under 2%, it could be considered that this calculation had converged. The relative error of every condition in the Table 4 are under 2%, and the mass flow follows the law of conservation of mass. The volumetric efficiency increases with the rotational speed, as shown in Figure 7.

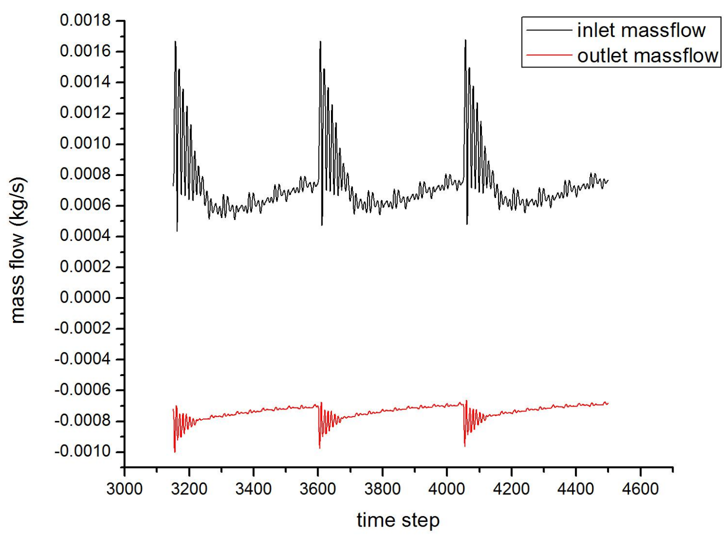


**Figure 6:** Mass flow

**Table 4:** Average flow

| Rotational Speed (rev/min) | Inlet Mass Flow (kg/s) | Outlet Mass Flow (kg/s) | Relative Error | Volumetric Efficiency |
|---|---|---|---|---|
| 4500 | 0.000166 | 0.000163 | 1.84% | 4.713% |
| 6000 | 0.000758 | 0.000752 | 0.77% | 15.93% |
| 8000 | 0.001862 | 0.001852 | 0.58% | 29.40% |

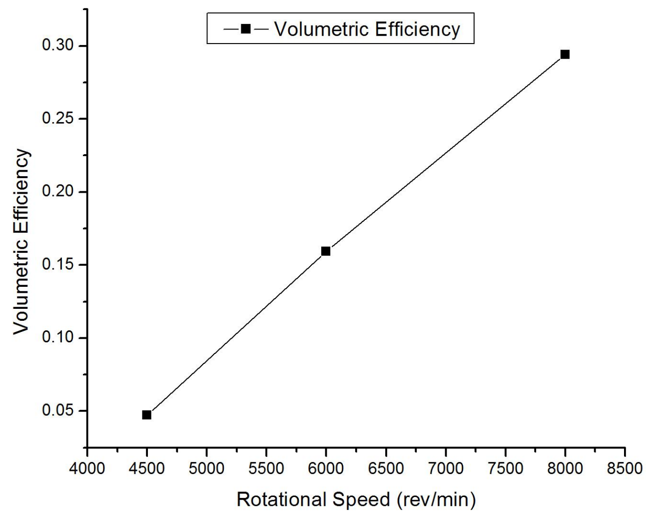


**Figure 7:** Volumetric efficiency vs. rotational speed

5.1.3 Torque: Periodic change of the torque in 6000 rev/min is shown in Figure 8. Other conditions had the same trends. The last complete period was selected. The average torque and shaft power were calculated and are shown in the Table 5. The trend of the shaft power is shown in Figure 9. Table 5 shows that the torque increased with the increasing rotational speed. And it is the same as shaft power.

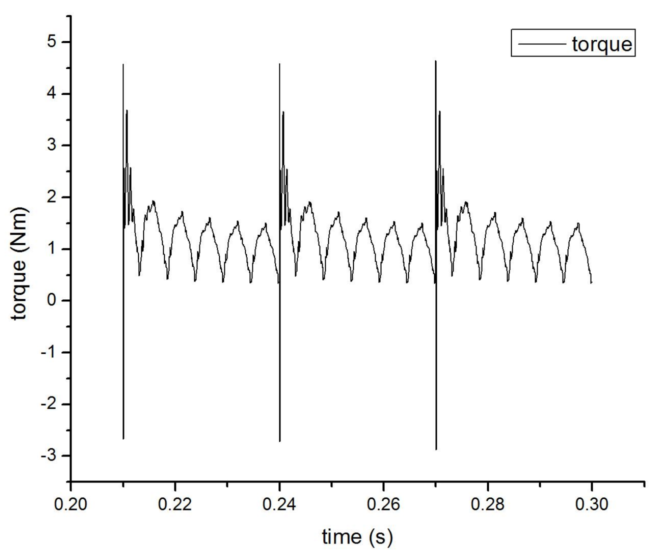


**Figure 8:** Torque

**Table 5:** Average torque and shaft power

| Rotational Speed (rev/min) | Torque (N·m) | Shaft Power (W) |
|---|---|---|
| 4500 | 0.948 | 447 |
| 6000 | 1.207 | 758 |
| 8000 | 1.224 | 1025 |

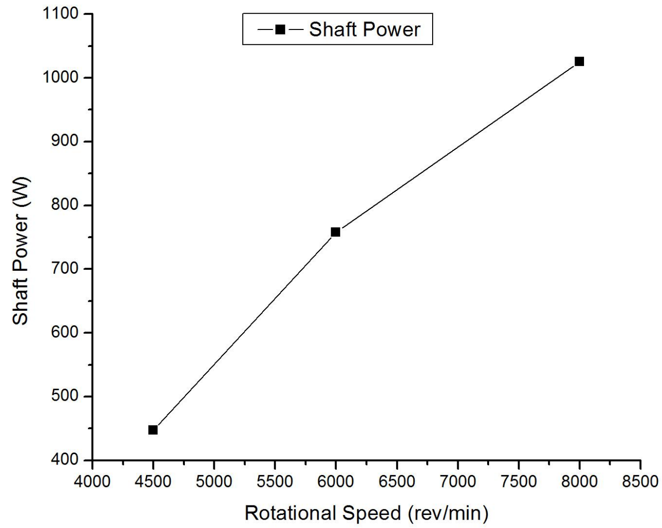


**Figure 9:** Shaft power vs. rotational speed

## 5.2 Different Mass Flows in Different Clearances

Clearance is an important factor that affects economy and reliability. From the perspective of economics, to reduce leakage and increase efficiency, it is better to have the smaller clearance. In terms of reliability, the static clearance taken should ensure that the dynamic gap must not be zero under any operating conditions. The actual clearance used must take into account some factors such as thermal expansion, deformation under load, clearances of the synchronization gears and bearings, processing and installation. It is difficult to determine accurately with theoretical calculation methods.

Two models, which had the same throttle diameters that the inlet throttle diameter was 4 mm and the outlet throttle diameter was 2.6 mm, were used to study the effect of different clearances. They had the same boundary conditions and initial conditions. The rotational speed was 6000 rev/min. The mass flows are shown in the Table 6. The two values of volumetric efficiency are 15.93% and 15.51%. The images of the flow fields in different rotation angles are shown in Figure 10. The left rotor rotated clockwise and the right rotor rotated counter-clockwise. The images show that the gas speed in the clearance was high and the gas leaked from the higher pressure chamber to the lower one, which led to the low efficiency. Table 6 also shows that the two models with different clearances have almost the same volumetric efficiency. Decreasing the clearances can lead to the decrease of the leakage area, however, which leads to the higher gas speed. These two factors effect each other, thus decreasing the clearances to the 0.05 mm of interlobe clearance and 0.03 mm of radial clearance is not very useful.

**Table 6:** The mass flow in two clearances

| Interlobe Clearance (mm) | Radial Clearance (mm) | Pressure Ratio | Inlet Mass Flow (kg/s) | Outlet Mass Flow (kg/s) | Volumetric Efficiency |
|---|---|---|---|---|---|
| 0.08 | 0.04 | 1.27 | 0.000758 | 0.000752 | 15.93% |
| 0.05 | 0.03 | 1.28 | 0.000738 | 0.000732 | 15.51% |

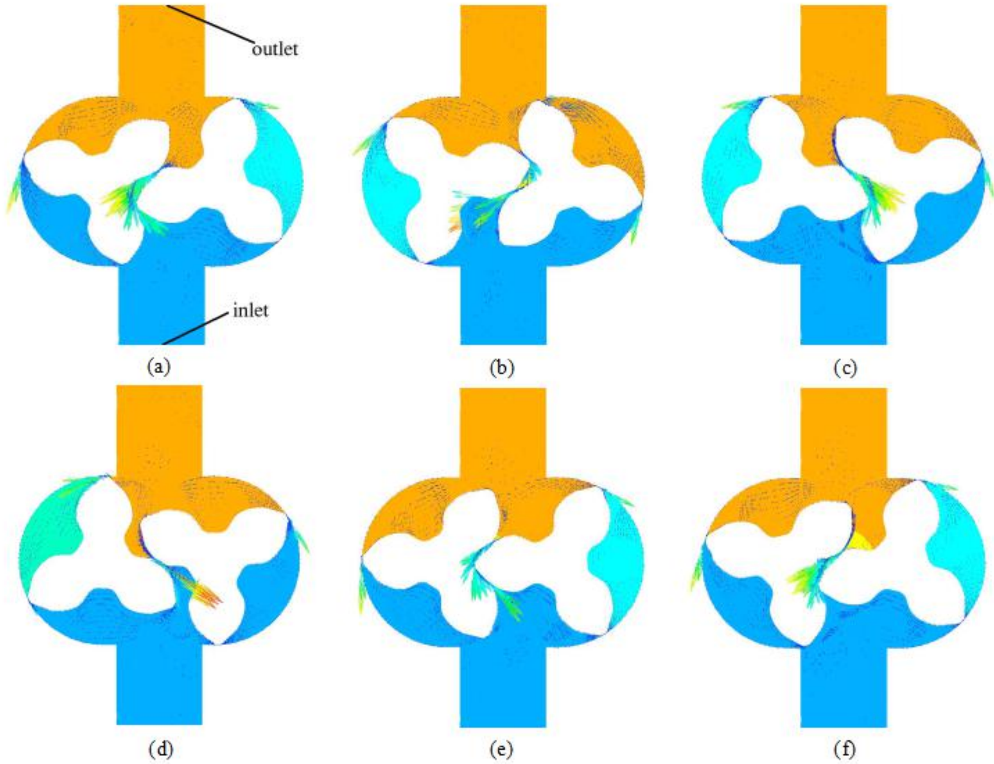


**Figure 10:** The flow fields of different angles

## 6. CONCLUSIONS

This paper shows CFD results for a roots blower using hydrogen as the working fluid used in FCV. Different rotational speeds of 4500 rev/min, 6000 rev/min and 8000 rev/min were applied, and the values of volumetric efficiency were 4.713%, 15.93% and 29.40%, respectively. The shaft powers were 447 W, 758 W and 1025 W. The hydrogen has a low relative molecular mass and is easy to leak. After the calculation of two different clearances, it is obvious that the clearances, the 0.08 mm of the interlobe clearance and the 0.04 mm of the radial clearance, are quite small. To decrease the clearances on the basis of the previous clearances has not much benefit. The effect of the increased gas speed and the effect of the reduced leakage area balance each other out. Further investigation will focus on the experimental verification and the strategy of promoting the volumetric efficiency. It is hoped that this study can offer some reference for the study of the full cell.

## REFERENCES


Song, P.P., Zhuge, W.L., & Zhang, Y.J. (2017). Unsteady Leakage Flow Through Axial Clearance of an ORC Scroll Expander. *Proceedings of the IV International Seminar on ORC Power Systems, Milano, Italy* (355-362).

Andres, R., Hesse, J., & Babic, H. (2016). CFD Simulation of a Twin Screw Expander Including Leakage Flows. *Proceedings of the 23rd International Compressor Engineering Conference at Purdue,* Purdue.

Hesse, J., & Andres, R. (2016). CFD Simulation of a Dry Scroll Vacuum Pump Including Leakage Flows. *Proceedings of the 23rd International Compressor Engineering Conference at Purdue,* Purdue.

Bianchi, G., Rane, S., & Kovacevic, A. (2017). Numerical CFD Simulations on a Small-scale ORC Expander Using a Customized Grid Generation Methodology. *Proceeding of the IV International Seminar on ORC Power Systems, Milano, Italy* (843-850).

Bianchi, G., Rane, S., & Kovacevic, A. (2017). Deforming Grid Generation for Numerical Simulations of Fluid Dynamics in Sliding Vane Rotary Machines. *Advances in Engineering Software*, 112(2017), 180-191.

Rane, S., Kovacevic, A., & Stosic, N. (2013). Grid Deformation Strategies for CFD Analysis of Screw Compressors. *International Journal of Refrigeration*, 36(2013), 1883-1893.

Suman, A., Randi, S., Casari, N. (2017). Experimental and Numerical Characterization of an Oil-Free Scroll Expander. *Proceedings of the IV International Seminar on ORC Power Systems, Milano, Italy* (403-410).
Fathabadi, H. (2018). Fuel cell hybrid electric vehicle (FCHEV): Novel Fuel Cell/SC Hybrid Power Generation System. *Energy Conversion and Management*, 156(2018), 192-201.
Hu, Y.S., Xu, R., & Huang, Y.F. (2017). Transient Analysis of Rolling Piston Compressor by CFD Method. *Fluid Machinery*, Vol.45, No.1, 2017. (In Chinese)
Chen, Y.K., Li, H.S., & Wu, T. (2016). Study on the Unsteady Flow of Gas in the Compressed Cavity of a Scroll Compressor. *Fluid Machinery*, Vol.44, No.10, 2016. (In Chinese)
Li, J.Q., Fang, C., & Xu, L.F. (2014). Current Status and Trends of the Research and Development for Fuel cell Vehicles. *J Automotive Safety and Energy*, 2014, Vol.5 No.1. (In Chinese)
Tao, W.Q. (2013). Grid Generation Techniques. In W.Q. Tao (Eds.), *Numerical Heat Transfer (Second Edition)* (432-482). Xian, China: Xian Jiaotong University. (In Chinese)